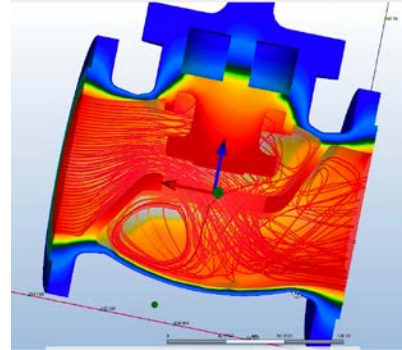


Lesson: Unsteady flow CFD analysis

In this lesson, you are introduced to unsteady flow and how it can affect other considerations in the design such as the thermal conditions of the model.

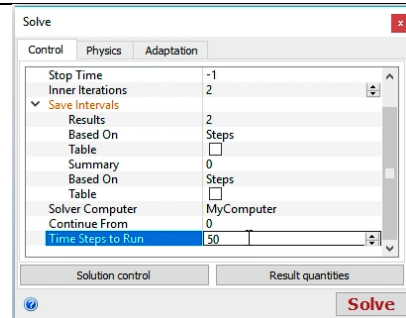
Learning Objectives:

- Set the criteria for an unsteady flow study.
- Run an unsteady flow study.
- Review the transfer of heat between bodies over the duration of the study.

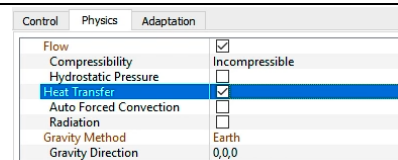


The completed exercise

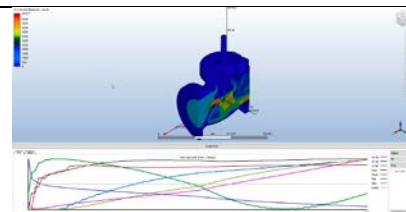
1. Continue with the file from the previous lesson. Click Setup>Solve. In the Solve panel, set the Solution mode to Transient and the Inner Iterations to 2. Under Save Intervals, set the Results to 2. Then Change the Time Steps to Run to 50.



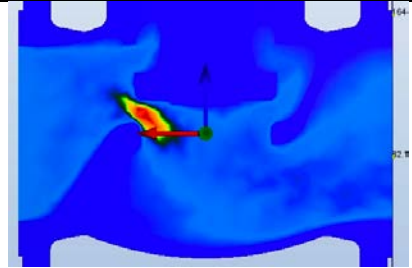
2. Navigate to the Physics tab and select Flow and Heat Transfer.



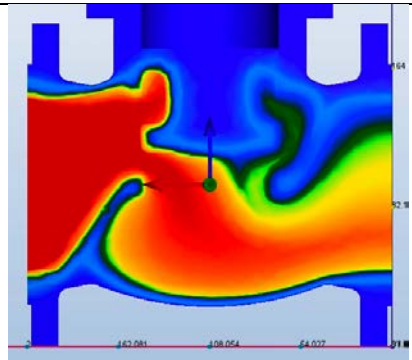
3. Click Solve and investigate the results. Click on the Output bar to reduce it which will open up some screen space.



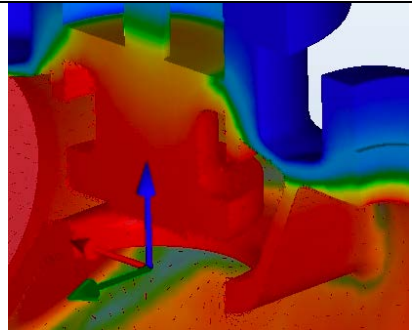
4. Use the Iteration/Step dropdown menu to note the changes that result from increasing velocity through the valve.



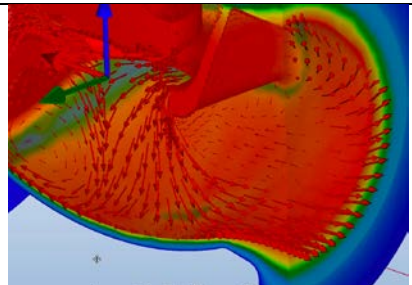
5. Change the Global Result to Temperature then review the Time Steps again to see how the temperature changes over time. The heat from the water is gradually transferred in to the body of the valve.



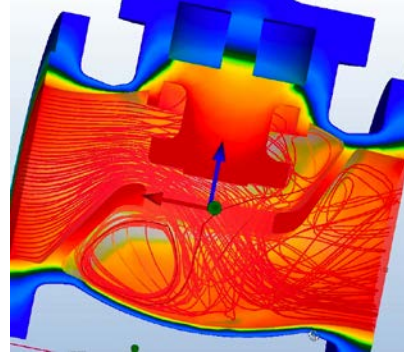
6. Right click on the fluid body and set it to Transparent. This allows you to see the structure of the valve body



7. Use the Global Vector dropdown menu to select Velocity Vector then click Vector Settings. Click Length range and change the Min value to 0.1 and the Max value to 0.35. The higher velocities are represented by longer arrows. Turn the Global Vectors off.



8. Change the appearance of the fluid body to be an outline. Click Results>Traces to put a circular grid on the model to show the flow as traces.



9. To share this data with others, click Results>Report Generator. Investigate the options then choose Generate. This will create an easily sharable Word document with the results.

